Copy No. 7

DEFENCE R&D DÉFENSE

Review of Detail Meshes for Application to Fatigue Analysis

D.R. Smith
Suite 707, 5959 Spring Garden Road, Halifax, NS

DEFENCE RESEARCH ESTABLISHMENT ATLANTIC

Contractor Report
DREA CR 2000-082
June 2000

National Defence Défense nationale

Canada

102-01436/

THIS IS AN UNEDITED REPORT ON SCIENTIFIC OR TECHNICAL WORK CONTRACTED BY THE DEFENCE RESEARCH ESTABLISHMENT ATLANTIC OF DEFENCE R&D CANADA

THE CONTENTS OF THE REPORT ARE THE RESPONSIBILITY OF THE CONTRACTOR, AND DO NOT NECESSARILY REFLECT THE OFFICIAL POLICIES OF THE DEPARTMENT OF NATIONAL DEFENCE.

PLEASE DIRECT ENQUIRIES TO:

THE DIRECTOR GENERAL
DEFENCE RESEARCH ESTABLISHMENT ATLANTIC
P.O. BOX 1012
DARTMOUTH, NOVA SCOTIA, CANADA
B2Y 3Z7

REPRODUCTION QUALITY NOTICE

This document is the best quality available. The copy furnished to DRDCIM contained pages that may have the following quality problems:

: Pages smaller or Larger than normal

: Pages with background colour or light coloured printing

: Pages with small type or poor printing; and or

✓ **: Pages with continuous tone material or colour photographs**

Due to various output media available these conditions may or may not cause poor legibility in the hardcopy output you receive.

☒ **If this block is checked, the copy furnished to DRDCIM contained pages with colour printing, that when reproduced in Black and White, may change detail of the original copy.**

Review of Detail Meshes for Application to Fatigue Analysis

D. R. Smith

D R Smith, Suite 707, 5959 Spring Garden Road, Halifax, N. S.

Defence Research Establishment Atlantic

Contract Report

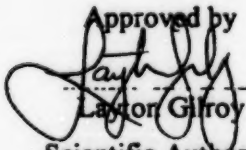
DREA CR 2000-082

June 2000

Author

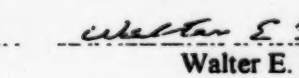
D R. Smith

Approved by



Layton Gilroy
Scientific Authority

Approved for release by



Walter E. Ellis
Chair, DREA Document Review Panel

© Her Majesty the Queen as represented by the Minister of National Defence, 2000
© Sa majesté la reine, représentée par le ministre de la Défense nationale, 2000

Abstract

Methods for determining the stress range in regions of high stress in ship structural details are described with emphasis on the hot spot method for fatigue analysis. Finite element models of typical details were created and analyzed to determine the hot spot stresses at the weld toe. Models of details with cut-outs were also analyzed by using essentially zero stiffness bar elements to determine the hot spot stresses at the plate edges of the cut-outs.

Résumé

Le présent document décrit les méthodes utilisées pour déterminer la plage des contraintes qui se produisent dans les zones de concentration de contrainte élevée de certains dtails de structure des navires, en mettant l'accent sur la mthode des points chauds pour l'analyse de la fatigue. Des modles à éléments finis de détails types ont été créés et analysés pour déterminer les contraintes de points chauds que l'on retrouve au pied des soudures. Des modéles de détails avec découpes ont aussi été analysés à l'aide d'éléments-barres d'une rigidité essentiellement nulle pour déterminer les contraintes de points chauds autour des découpes.

DREA CR 2000-082

Review of Detail Meshes for Application to Fatigue Analysis

by

D.R. Smith

Executive Summary

Introduction

The structural design of a ship hull is generally finalized by a global finite element analysis of the structure. The results of the global analysis can be used to identify the regions of nominal high stress in the hull structure. An additional step is then required to examine the detail structure in the high stress regions whereby a portion of the global model is extracted for further analysis. The extracted portion is refined in detail to model the beams and the girders and their intersection with frames and bulkheads. The details of the beam connections are often weldments of such complexity that individual connections together with a portion of the surrounding structure must be modeled in still greater detail with a second level of extraction to obtain the the desired accuracy in stress results. This second level of analysis is most necessary when the structural fatigue of connections is to be assessed. In this manner, the stress at points of possible fatigue due to cyclic loading can be obtained for a fatigue analysis. In a single cycle the difference between the maximum and minimum stress is the stress range. It is used with the appropriate S-N curve to predict the number of cycles to failure.

The stress can be calculated by four methods which are the nominal stress approach, the fine finite element grid or fine mesh approach, the notch stress approach, and the hot spot stress approach. They are basically used to determine stresses at points of maximum stress concentration that occur at the edges of holes, cutouts and the toe of welds (intersection of the weld with the plate) which under cyclic loading can lead to fatigue cracks.

This report demonstrates the fine grid finite element approach but mainly concentrates on the hot spot approach which is applied to a number of geometries found in ship structural details ranging from holes and cutouts to weldments. The geometries are meshed with plate and solid elements, using various degrees of finite element mesh refinement, to demonstrate the method. Also demonstrated is the use of essentially zero stiffness bar finite elements to identify and determine hot spot stresses at hole and cutout edges.

Principal Results

Results were obtained from the fine grid finite element analysis of a weld toe that indicated final convergence of the stress to a reliable value from which the stress concentration for the weld toe was calculated.

A structural detail listed in the catalog of the American Institute of Steel Construction was modeled with finite elements. The hot spot stress results showed that plate element models were simpler to use than the solid elements and gave good results. The solid element model allowed the weld to be included in the model. It gave good results, comparable with the plate element models when the weld was included, but performed poorly when it was not.

The use of bar elements to obtain hot spot stresses, in a plain plate with a hole, was tested with a variety of element meshes. Though all elements performed well, the 4-node plate element gave the best results, giving a stress concentration factor in excellent agreement with published values. The bar elements were also shown to perform well in predicting hot spot stresses for cutouts.

A weldment, in the form of a ship bracket detail whose finite element analysis results had been published, was chosen as benchmark test of the hot spot method. The bracket was modeled using 4-node plate elements. Bar elements were used at the edges and the stresses in bar and the edge plate elements were obtained. The hot spot stress results, obtained using VAST, were in reasonable agreement with the published results for the plate elements, but varied considerably for the bar elements.

Significance of Results

The review of finite element meshes for typical structural details occurring in ship structures to obtain hot spot stresses for use with S-N curves showed, that in most cases, the 4-node plate element was the best choice for finite element modeling. The 20-node solid element provided the ability to model the weld, and when used for this purpose, it gave good results. The use of essentially zero stiffness bar elements at the edges of cutouts and holes is a reliable and accurate method for obtaining hot spot stresses. The comparison of the ship bracket VAST results with the ANSYS results showed a disappointing lack of agreement between the two analyses which may have been due to a difference in element performance in the programs.

CRDA DCD 2000-082

Étude du maillage des détails et de son application à l'analyse de la fatigue

par

D.R. Smith

Sommaire

Introduction

La dernière étape de la conception d'une coque de navire consiste habituellement à effectuer une analyse par lments finis globale de sa structure. Les résultats de lanalyse globale peuvent alors servir à identifier les zones de contrainte nominale élevée dans la structure de la coque. Pour tudier un détail de structure dans une zone de contrainte éleve, il faut passer à une étape supplémentaire qui consiste à extraire une portion du modèle global et à en faire une analyse plus poussée. Pour ce faire, on réalise un modèle finement détaillé des poutres et des supports de la portion extraite et de leur intersection avec les membrures et les cloisons. Ces raccordements de poutres prennent souvent la forme dassemblages souds tellement complexes quil faut en modéliser les connexions individuelles et une portion de la structure avoisinante avec encore plus de détail par le biais dun deuxième niveau dextractions pour obtenir des valeurs de contrainte de la prcision voulue. Ce deuxième niveau danalyse savère d'autant plus nécessaire quand il faut évaluer la résistance à la fatigue des connexions. Cette méthode permet de faire une analyse de la résistance la fatigue partir des contraintes mesures aux points susceptibles la fatigue parce que soumis des charges cycliques. Lécart entre les contraintes limites suprieure et infrieure mesurées au cours dun seul cycle sappelle la plage des contraintes. De concert avec la courbe de fatigue, elle sert à prédire le nombre de cycles à la rupture.

La contrainte peut être calculée à laide de l'une des quatre méthodes suivantes : la méthode des contraintes nominales, la méthode des modles à élments finis ou à mailles fines, la mthode de la concentration des contraintes sur éprouvette entaillée et la méthode de la concentration des contraintes aux points chauds. Toutes ces méthodes servent essentiellement à calculer les contraintes dans les zones de concentration maximale, à savoir autour des trous, des découpes et au pied des cordons de soudure (à lintersection de la soudure et de la tôle), qui sont les plus susceptibles à la fissuration par fatigue quand elles sont soumisesà des charges cycliques.

Le présent rapport décrit la méthode des éléments finis à mailles fines, mais il se penche surtout sur la méthode de la concentration des contraintes aux points chauds appliquée aux diffrentes formes que l'on retrouve sur les détails de structure d'un navire, notamment aux trous, aux découpes et aux assemblages soudés. Le maillage des formes avec des éléments finis ayant la forme de plaques (lments-plaques) et de solides est ensuite réalis, à divers degrés de raffinement des éléments finis, pour dmontrer le fonctionnement de la méthode. Le rapport démontre aussi comment utiliser un élément-barre (élément fini ayant la forme d'une barre) d'une rigidité essentiellement nulle pour identifier et calculer les concentrations de contrainte à un point chaud, comme par exemple autour d'un trou ou d'une découpe.

Principaux résultats

Les résultats de lanalyse par éléments finis à mailles fines du pied dun cordon de soudure indiquent que les valeurs de convergence dfinitive des contraintes sont assez fiables pour servir au calcul de la concentration des contraintes pour un pied de cordon de soudure.

Un détail de structure inscrit au catalogue de l'American Institute of Steel Construction a été modélisé par la méthode des éléments finis. Les mesures de contrainte aux points chauds obtenues par cette méthode dmontrent que les modéles à élément-plaque sont plus faciles à utiliser que les éléments finis solides et donnent de meilleurs rsultats. Le modle à élément solide permet d'incorporer la soudure au modéle. Il donne de bons résultats, comparables à ceux des modles à éléments-plaques quand la soudure y est incorporée, mais beaucoup moins bons quand elle ne l'est pas.

On a mis à l'épreuve des éléments-barres, en y maillant divers éléments, pour tenter dobtenir des mesures de contraintes aux points chauds. Bien que tous les éléments essays aient eu un bon rendement, c'est l'élément-plaque à quatre noeuds qui a donné les meilleurs résultats, produisant un facteur de concentration des contraintes en accord avec les valeurs publiées. Les éléments-barres ont aussi réussi à prédire correctement les contraintes aux points chauds des découpes.

Des essais sur un assemblage soudé (de la forme dun gousset de navire pour lequel des résultats danalyse par éléments finis avaient été publiés) ont été choisis comme point de référence pour la méthode de la concentration des contraintes aux points chauds. Le gousset a été modélisé à l'aide déléments-plaques à quatre noeuds. Des éléments-barres ont été utilisés sur les arêtes pour calculer les contraintes subies par les éléments-barres et larête des éléments-plaques. Les résultats obtenus en matire des contraintes aux points chauds à laide du logiciel VAST étaient suffisamment conformes aux rsultats publiés à propos des éléments-plaques, mais étaient fort dissemblables en ce qui concerne les éléments-barres

Signification des résultats

L'étude de modles à éléments finis de détails types de la structure des navires en vue de mesurer les contraintes aux points chauds et de combiner ces dernières aux courbes de fatigue a démontr que, dans la plupart des cas, l'élément-plaque à quatre noeuds s'avère le meilleur choix pour la modélisation par éléments finis. L'élément solide à vingt nuds permet de bien modéliser la soudure et donne des bons rsultats quand on l'utilise à cette fin. L'utilisation d'éléments-barres d'une rigidité essentiellement nulle autour des découpes et des trous s'est avérée une méthode fiable et précise de mesurer les contraintes aux points chauds. Il est décevant de constater que les résultats de l'étude d'un gousset de navire avec le logiciel VAST ne concordent pas avec ceux du logiciel ANSYS mais il se peut que les diffrences soient causes par un cart de rendement des divers éléments des programmes en question

Contents

Abstract ii

Executive Summary iii

Table of Contents vii

List of Figures viii

List of Tables ix

1 Introduction 1

2 Nominal Stress Approach 3

3 Fine Grid Finite Element Approach 4

4 Notch Stress Approach 5

5 The Hot Spot Stress Approach 6

6 Finite Element Modeling for Hot Spot Stresses Determination 8
- 6.1 Lateral Attachments to a Plate 8
- 6.2 Plain Plate with Hole 11
- 6.3 T Section with a Cut-out 14
- 6.4 A Ship Bracket Detail 16
- 6.5 Cut-out for Stiffeners 17

7 Summary and Recommendations 19

References 21

List of Figures

1 MAESTRO Global Finite Element Model 1
2 Portion to be Extracted 2
3 Extracted Portion of Hull Structure 2
4 Refined Model of Extracted Portion 2
5 Second Extraction and Refinement 2
6 Nominal Stress and Local Geometric Stress Distribution 4
7 AISC S-N Curves for Welded Joints 5
8 TWI S-N Curves for Welded Joints 5
9 Finite Element Model of Fillet Weld 5
10 Refined Weld Toe with 1/32 in Radius 5
11 Definition of Nominal, Hot Spot and Peak Stress 7
12 The Use of Bar Elements to Obtain Hot Spot Stresses at a Cut-out 7
13 Hot Spot Stress in the 4 Node Plate Model of Detail 30 Category E 9
14 Hot Spot Stress in the 8 Node Plate Model of Detail 30 Category E 9
15 Hot Spot Stress in the 20 Node Solid Model of Detail 30 Category E 9
16 Hot Spot Stress in the 25 mm square 8 Node Plate Model of Detail 30 Category E 10
17 Hot Spot Stress in the 8 Node Model of Detail 30 Category E with 5 mm Plate . 10
18 Hot Spot Stress in the 20 Node Solid Model of Detail 30 Category E Including the Welds 11
19 4 Node Plate and Bar Element Stresses in a Plain Plate with a Hole Under tension 12
20 8 Node Plate and Bar Element Stresses, in a Plain Plate with a Hole under Tension 12
21 20 Node Plate and Bar Element Stresses, in a Plain Plate with a Hole under Tension 13
22 20 Node Plate and Bar Element Stresses, in a Plain Plate with a Hole under Tension using a Finer Grid 13
23 4 Node Plate and Bar Element Stresses in a T section with a Cut-out under Tension 14
24 20 Node Solid and Bar Element Stresses in a T section with a Cutout under Tension 15
25 4 Node Plate and Bar Element Stresses in a Refined T section with a Cutout under Tension 15
26 4 Node Plate and Bar Element Stress Distribution in a T section with a Cutout under a Tension and Transverse Load 16
27 The Ship Bracket Finite Element Model with Bar Sizes and Overall Dimensions . 17
28 Von Mises Stresses in the Bracket and an Expanded View Showing the Element and Bar Stresses 18
29 Hot Spot Stress in the Beam Web 18
30 Hot Spot stress in the Bracket Gusset 18
31 Von Mises Stresses in the Bracket with Better Proportioned Elements and an Expanded View Showing the Element and Bar Stresses 19
32 Hot Spot Stresses in a Cut-out Obtained by the use of Bar Elements 19

List of Tables

1 Maximum Ist Principal Element Stress due to Tension in Detail 30 Category E, Models 1 to 6 11
2 Max Element and Hot Spot Stresses in a Plain Plate with a Hole, Models 1 to 4 14
3 Max Element and Hot Spot Stresses in a T Section with Cut-out 16
4 A Comparison of Von Mises Element and Bar Stresses in a Ship Bracket Detail . 17

1 Introduction

The structural design of a ship hull is generally finalized by a global finite element analysis of the structure using a program such as MAESTRO[1]. It is at this stage that the scantlings are optimized for the expected loading conditions. The finite element model for such an analysis as shown in Figure 1 is, by necessity, lacking in detail in the modeling of the connections such as the intersection of longitudinals with transverse frames, webs, and bulkheads.

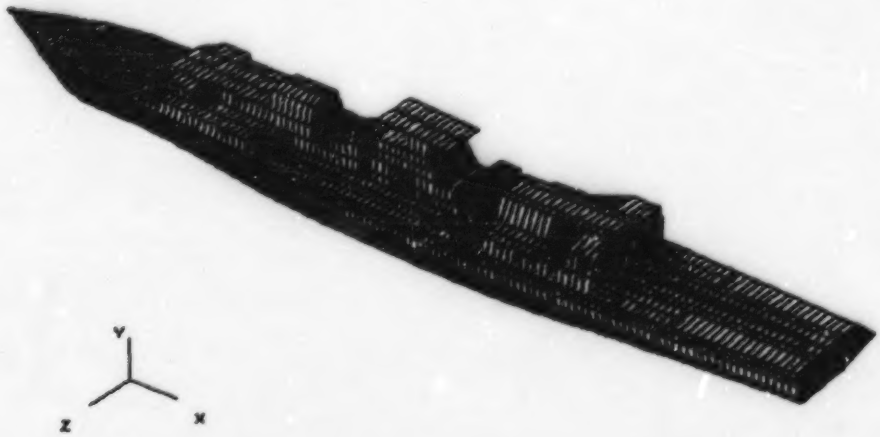


Figure 1: MAESTRO Global Finite Element Model

The results of a global analysis using such a model can be used to identify the regions of nominal high stress in the hull structure as shown in Figure 2. An additional step is then required to examine the detail structure in the high stress regions. This can be achieved by a top-down analysis, for which the program MG/DSA[2] is ideally suited, whereby a portion of the global model is extracted, as shown in Figure 3.

The extracted portion is refined in detail to model the beams and girders and their intersection with frames and bulkheads Figure 4. The boundary conditions for the detail model are obtained from the global analysis and are applied to the boundaries of the refined model. Loads including pressure loads are also applied where necessary. The details of the beam connections are weldments of such complexity that individual connections together with a portion of the surrounding structure must be extracted and modeled in still greater detail, as shown in Figure 5, to obtain the desired accuracy in stress results.

This second level top-down analysis is most necessary when the structural fatigue of the connections is to be assessed. In this manner, the stress ranges σ_r at points of possible fatigue failure due to the cyclic loading can be obtained for a fatigue analysis. The stress range σ_r, in a single stress cycle, is equal to σ_{max} minus σ_{min}. It is then used with the appropriate S-N curve for the detail, an experimentally obtained log-log plot of the cycles-to-failure versus stress, to predict the number of cycles-to-failure. As the cyclic stresses are due to variable amplitude loading, the fatigue damage is then assessed by the use of the Palmgren-Miner's[3] cumulative

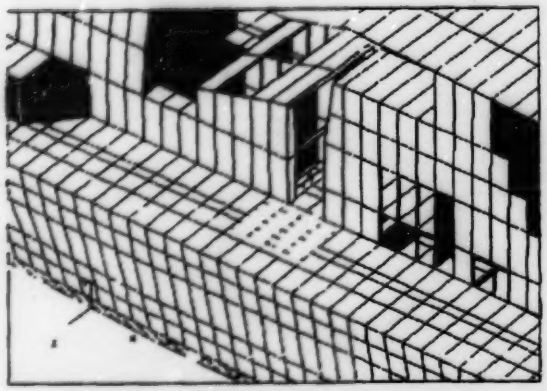


Figure 2: Portion to be Extracted

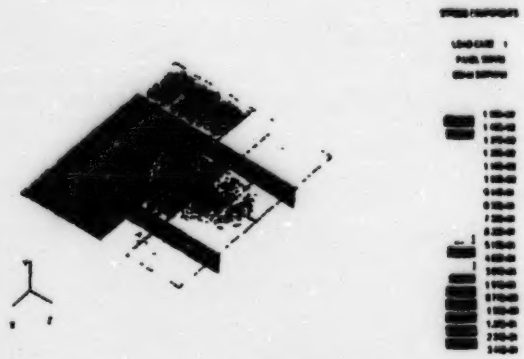


Figure 3: Extracted Portion of Hull Structure

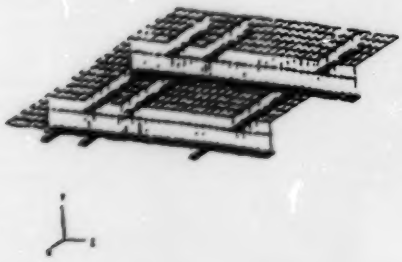


Figure 4: Refined Model of Extracted Portion

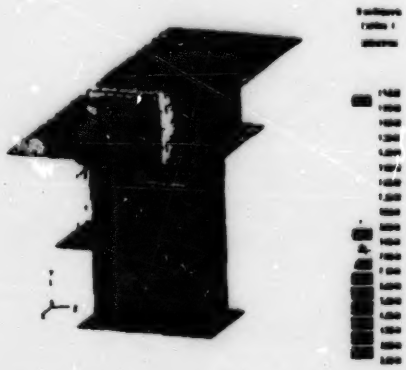


Figure 5: Second Extraction and Refinement

damage rule. Using this rule, the ratio of the number of cycles n_i at a stress range i to the number of cycles to failure N_i at stress i are summed for the number of different stress levels k. The sum D must be less than 1.

$$D = \sum_{i=1}^{k} \frac{n_i}{N_i} =\leq 1$$

The determination of the values of the stress ranges is therefore fundamental to the fatigue analysis of structural details. The stress can be calculated by four methods which are the nominal stress approach, the fine finite element grid approach, the notch stress approach, and the hot spot stress approach. They are basically used to determine stresses at points of maximum stress concentration, which experience has shown occur at the toe of welds. It is at the weld toe that fatigue cracks are most likely to start.

2 Nominal Stress Approach

Provided the structural detail and the loading are not too complex, combined axial and bending stresses can be calculated from the equation[4]:

$$\sigma_n = \frac{P}{A} + \frac{Mc}{I}$$

where

σ_n = the nominal stress
P = axial force
A = cross section area
M = moment
c = offset from the neutral axis
I = moment of inertia

These so called nominal stresses may be complicated by such things as shear lag and anticlastic curvature. These and other effects for which there are well established stress analysis methods should be accounted for if they are determined to be significant. Once the nominal stress σ_n has been determined, it is used in the following equation to obtain the peak stress range at the weld toe.

$$\sigma_r = \sigma_n K_g K_w$$

where

σ_r = the peak stress range
σ_n = nominal stress
K_g = geometric stress concentration
K_w = weld shape stress concentration

Both K_g and K_w can be selected from published tables for typical structural weldments such as Table 7.1 and Table 7.2 found in Reference[5] provided the detail is similar to one of the geometries shown in the tables.

In the situation where the detail and loading are complicated, the nominal stress may be determined by a finite element analysis in which the detail is extracted from the global model and

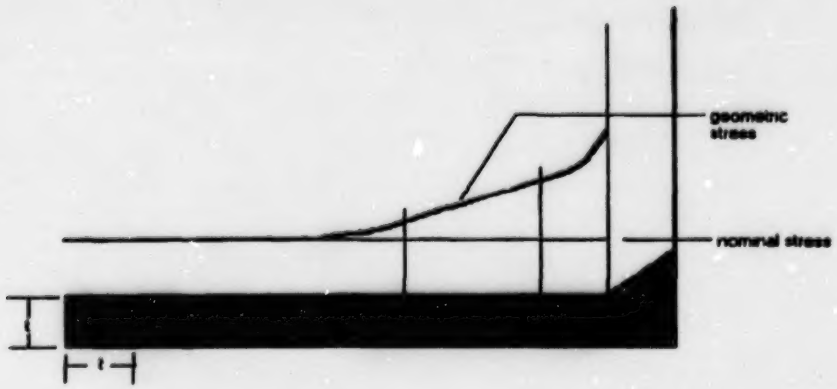


Figure 6: Nominal Stress and Local Geometric Stress Distribution

refined. The boundary conditions along with the loading are automatically obtained and applied in the extraction process. In this case the nominal stress, as shown in Figure 6, is determined at a location free from the effects of local geometry and the weld stress concentration. The stress concentration factors are chosen in the same manner as before.

The nominal stress method has progressed to a third stage. It is based on a large data base obtained from the fatigue testing of weldments commonly found in ship structures[6]. The resulting S-N data has been categorized by a series of curves[7] from A to G as shown in Figure 7. Other series of S-N curves for welded joints have been produced by the UK Department of Energy as illustrated in Reference[8] and by the UK Welding Institute as shown in Figure 8. Because these curves account for the K_g and the K_w stress concentration factors they are chosen on the basis of their similarity to the detail or portions of the detail being analyzed and can be used directly with the nominal principal stress normal to the weld toe.

3 Fine Grid Finite Element Approach

A second method for determining the peak stress at the weld toe in a ship structural detail is by a fine grid finite element analysis with increasing grid refinement in the region of the weld toe. An analysis of this type was carried out using the program VAST[9] with a finite element model of 3500 8-node shell elements. The model and results are shown in Figure 9 and Figure 10. This approach is time consuming and is further complicated by the presence of a singularity at the intersection of the weld with the plate requiring the use of a small radius of 1 mm or 1/32 in. to achieve convergence. The method, however, has the added value of providing the ability to calculate the overall stress concentration factor K_t at the weld for use in the analysis of similar structural details. Therefore:

$$K_t = \frac{\sigma_r}{\sigma_n}$$

From Figures 9 and 10, where a 1/32 inch radius was used to obtain convergence, the nominal

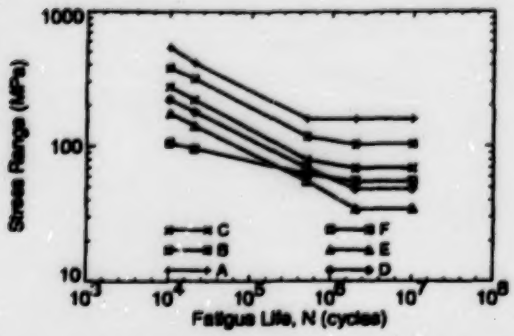


Figure 7: AISC S-N Curves for Welded Joints

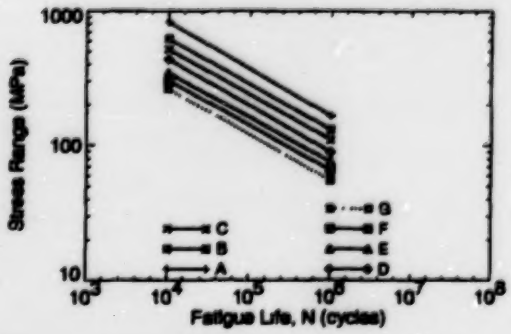


Figure 8: TWI S-N Curves for Welded Joints

stress was 1000 psi and the stress at weld toe radius was 3270 psi. The stress concentration factor is:

$$K_t = \frac{3270}{1000} = 3.27$$

The S-N curve for this approach is that obtained for the material using smooth standard fatigue test specimens. An additional stress concentration factor of 1.4 should be applied to allow for the plate roughness.

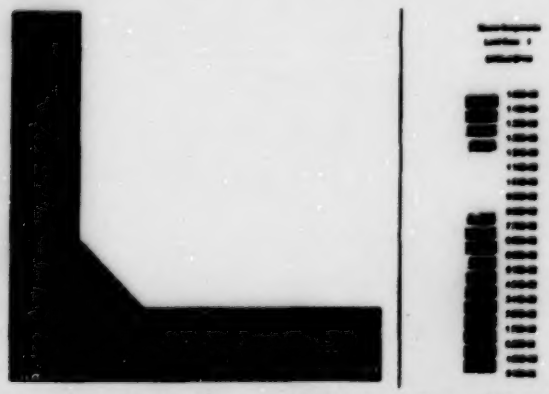


Figure 9: Finite Element Model of Fillet Weld

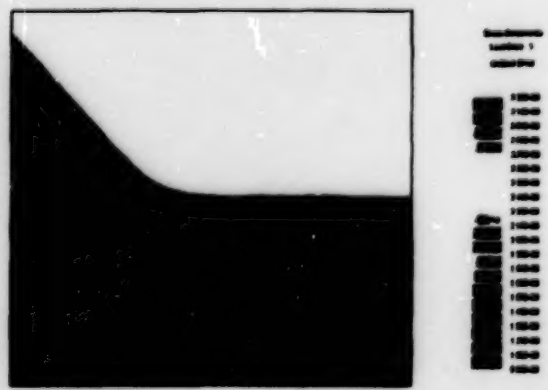


Figure 10: Refined Weld Toe with 1/32 in Radius

4 Notch Stress Approach

This method is based on a similar approach to one used in machine design where the nominal stress is multiplied by a stress concentration factor due to a localised geometric effect. This works well for notch geometries such as semi-circular edge notches in plates for which there are published stress concentration factors. The method has now been used to include a notch

factor due to the local geometry of the weld toe assuming a notch radius of 1 mm The notch affect has been found to be sensitive to plate thickness[7].

5 The Hot Spot Stress Approach

A fourth method, which was initially used by for fatigue analysis of tubular joints of offshore structures, is known as the "hot spot" stress approach. It is applied mainly to predicting fatigue failure that may occur at the toe of the welds in a structural detail where the welds are transverse to the alternating stress component. It is best applied to plate thicknesses no greater than 1 inch or 25 mm. As stated in Reference[4] the hot spot approach is preferable to the nominal stress approach when:

(a) the nominal stress is not clearly defined
(b) the detail cannot be identified amongst those for which S-N curves are available
(c) because of the detail complexity the finite element method must be used
(d) stresses are to be checked with strain gauge measurements

The hot spot method uses a finite element analysis of the structural detail to obtain membrane and bending stresses which are mainly linear through the plate thickness and normal to the weld toe but far enough from the weld toe to be free of the effect of the weld. Two stress values are chosen at the recommended[5] distances of t/2 and 3t/2 normal to the weld toe, where t is the plate thickness, for a linear extrapolation of the stress to the weld toe, as shown in Figure 11. The hot spot stress at this stage includes the geometric stress concentration but not the weld stress concentration K_w. There are a number of options at this point. The value for K_w can be obtained from published tables or calculated from the following equation[8].

$$K_w = 1.6\left(\frac{\theta}{30}\right)^{\frac{1}{2}}$$

where θ is the mean weld toe angle in degrees.

The hot spot stress can multiplied by K_w to obtain the peak stress range value. The peak stress value however should not be used with the S-N curve for the weld type used as it has already been accounted for in the S-N data. It may be used with the basic S-N curve for the material.

The preferred option however is to apply the hot spot directly to an S-N curve which has the weld concentration included, such as the S-N curved defined for all tubular joints used in offshore structures[10]. This presents a problem for other types of connections, as there are as many as 9 curves proposed by different classification societies[11] for fillet welded joints. The stress range of these curves vary from 450 to 600 Mpa on their intercept with the vertical axis at 10^4 cycles. There is also the question as to whether the stress concentration due to a particular type of weld is accounted for in the chosen curve. The E curve in the series of S-N curves of Figure 7 is considered suitable for hot spots at fillet welds.

Another method for determining hot spot stresses is through the use of essentially zero stiffness bar elements. This method is applied mainly to cut-outs where no geometric singularity exists, as shown in Figure 12.

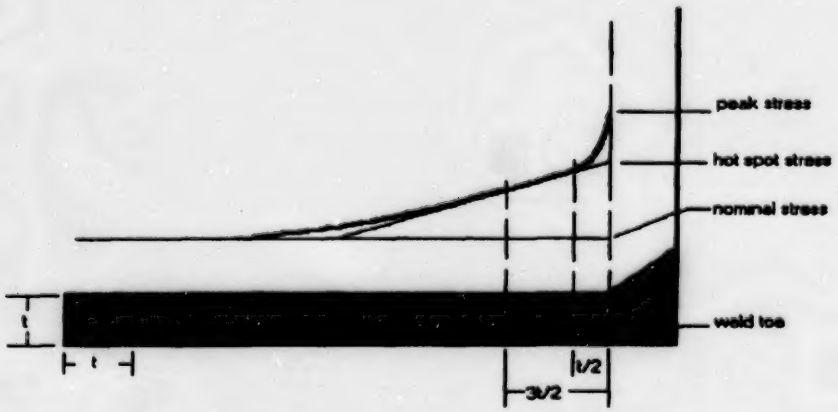


Figure 11: Definition of Nominal, Hot Spot and Peak Stress

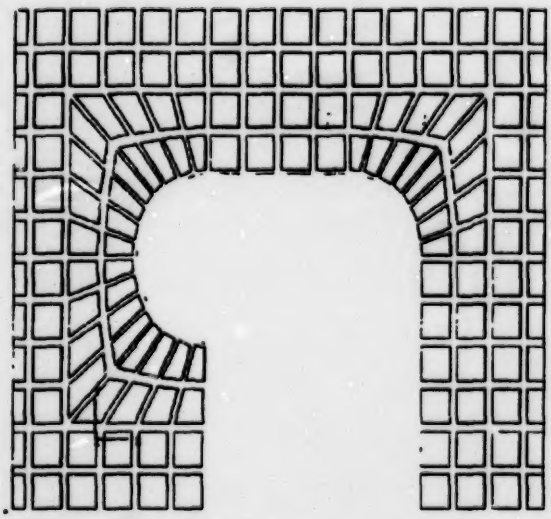


Figure 12: The Use of Bar Elements to Obtain Hot Spot Stresses at a Cut-out

6 Finite Element Modeling for Hot Spot Stresses Determination

In modeling to determine hot spot stresses in a structural detail, two approaches are used. One approach is not to include the welds in the model. This has the advantage that shell elements, as well as 20-node solid elements, can be used. The second approach is to include the weld in the model. This approach requires the use of 20-node solid elements and presents more difficulty in modeling. It therefore is desirable in most cases to use 4 or 8-node shell elements to model the structural detail which results in a simpler model and eliminates the problem of including the weld. Without the weld present it is recommended in Reference 5 that the stress points required for the linear extrapolation be chosen at half the plate thickness and 1.5 times the plate thickness from the intersection of the elements. This rule also applies if solid elements are used. It works well when the stresses are predominately membrane stresses, which are uniform through the plate thickness. It is also recommended that the element sizes be equal to the plate thickness for the 4-node and 20-node elements. The sides of the 8-node elements should be equal to twice the plate thickness. This rule for element size cannot always be met exactly because of the proportions of some structural details.

6.1 Lateral Attachments to a Plate

A test of the hot spot method was made using structural detail 30 of category E of the AISC fatigue provisions[6]. The dimensions chosen were those of a fatigue test specimen[12]. Initially four finite element models were created without the weld. The first was created using 4-node plate elements and 8-node plate elements were used to create the second. The third was created using 20-node solid elements. The plate thickness of the detail was 12 mm For ease of modeling the elements lengths and widths were initially set to 10 mm. In the case of the 8-node plate element model, a fourth model was created with the lengths of the element sides set to 25 mm to more truly meet the 2t rule. The models were placed under a tension load. The results for the first model for the 4-node plate model are shown in Figure 13 together with a plot of the extrapolation of the principal stress normal to the weld toe to obtain the hot spot stress. The results for the 10 mm x 10 mm 8-node plate element model are shown in Figure 14 and the 20-node solid model results are shown in Figure 15.

The results of the 25 mm square 8-node element model are shown in Figure 16. Because the stress for this model, based on the 2t rule for 8-node plate elements, produced maximum stresses that were considerably lower than the other models it was decided to create a model that met the criteria exactly. This was achieved by reducing the plate thickness from 12 mm to 5 mm in the 10 mm square 8-node plate element model. The loading was adjusted to maintain the same nominal stress. The results for this model are shown in Figure 17.

A sixth model was created of the detail with the weld modeled with 20-node solid element degenerated into wedges. Ten node tetrahedrons were used for the corners. The results of the analysis are shown in Figure 18.

The results of the 6 analyses are summarized in Table 1. They showed reasonable agreement between the 4 and 8-node elements when modeled with 10 mm sized elements. The results obtained when modeled with 25 mm 8-node elements, which more closely obeys the 2t rule, were much lower at 9.85 MPa. When the thickness was changed to 5 mm, and 8-node 10 mm elements were used, the same stress of 11.9 resulted as when the 10 mm thickness was used showing that the 8-node element is more sensitive to grid size than element size.

The 20-node element model, with 10 mm elements, gave a much higher stress than the plate

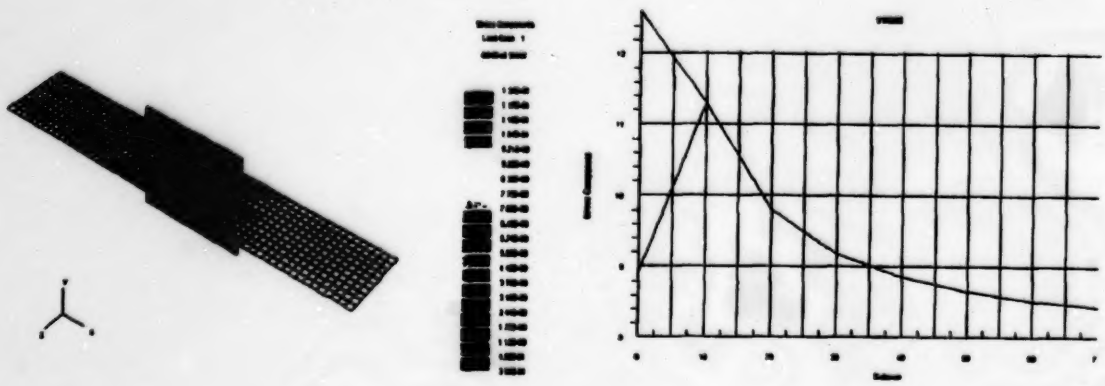


Figure 13: Hot Spot Stress in the 4-Node Plate Model of Detail 30 Category E

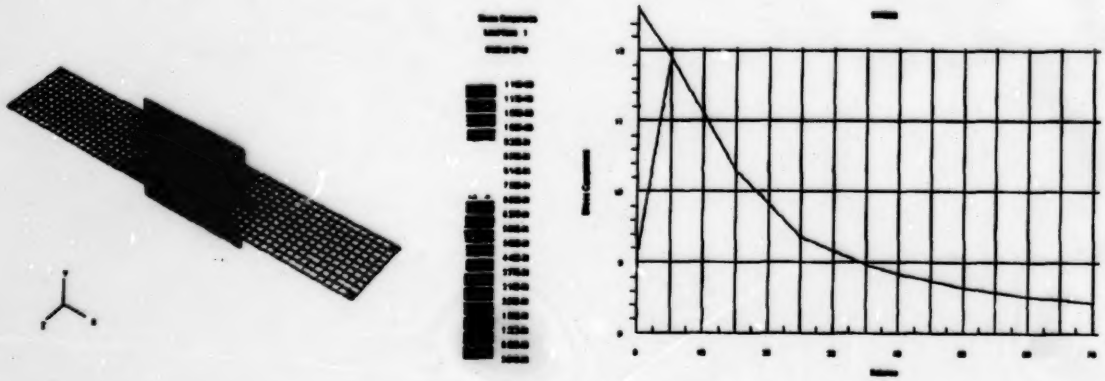


Figure 14: Hot Spot Stress in the 8-Node Plate Model of Detail 30 Category E

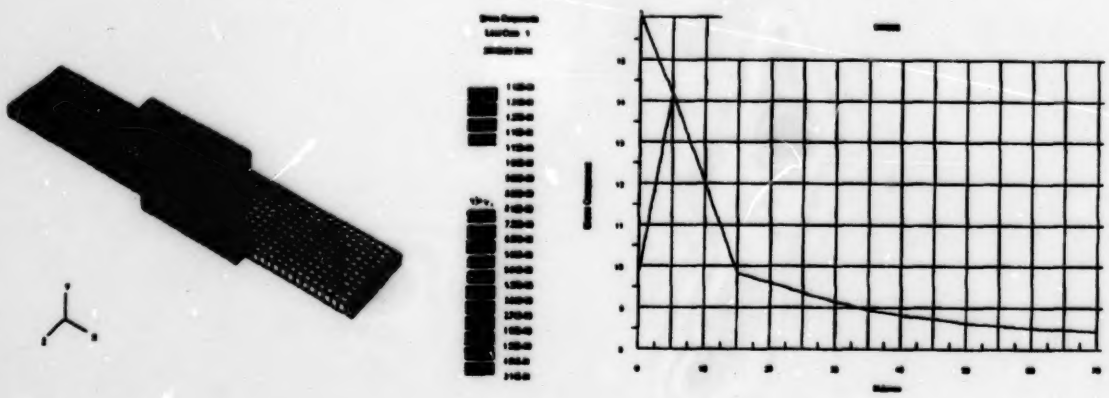


Figure 15: Hot Spot Stress in the 20-Node Solid Model of Detail 30 Category E

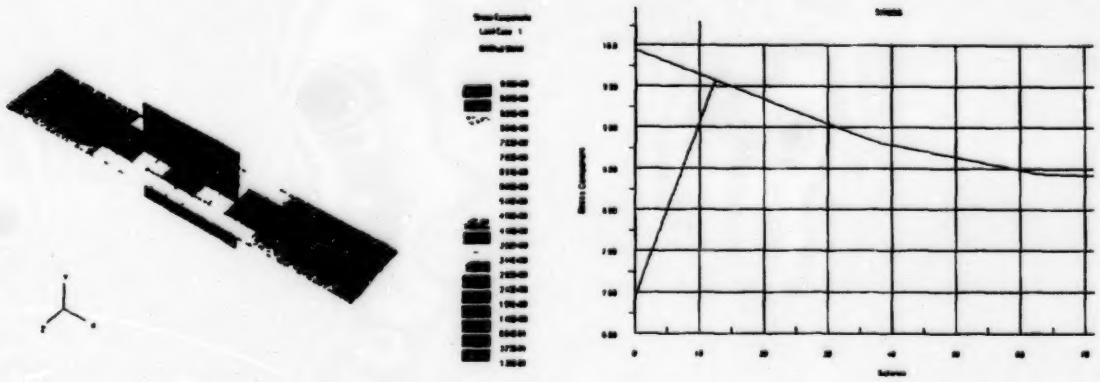


Figure 16: Hot Spot Stress in the 25 mm square 8-Node Plate Model of Detail 30 Category E

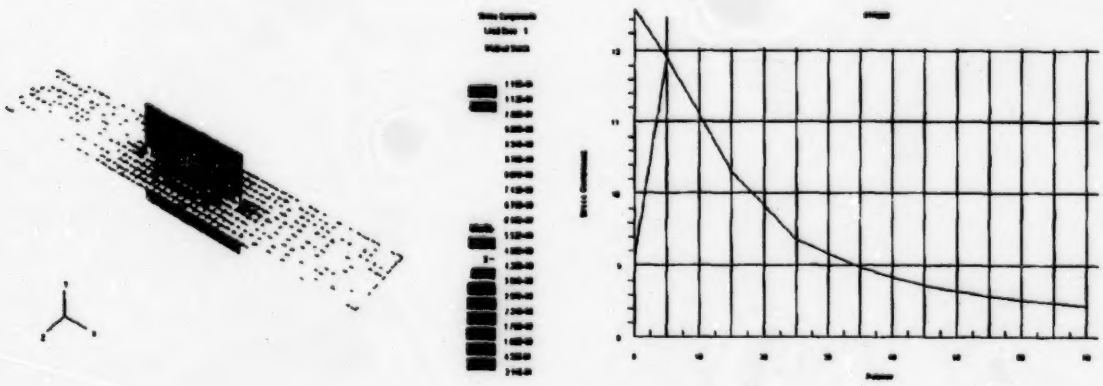


Figure 17: Hot Spot Stress in the 8-Node Model of Detail 30 Category E with 5 mm Plate

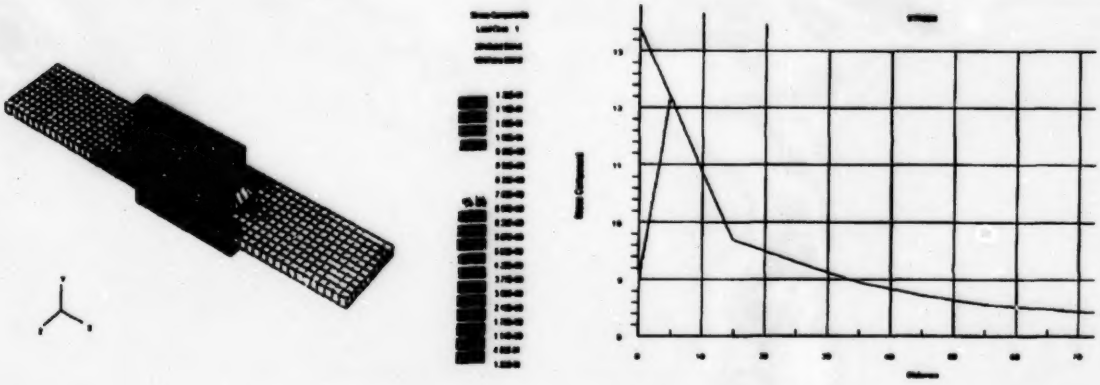


Figure 18: Hot Spot Stress in the 20-Node Solid Model of Detail 30 Category E Including the Welds

Table 1: Maximum Ist Principal Element Stress due to Tension in Detail 30 Category E, Models 1 to 6

Model	Element Type	Element Size	Weld Modeled	Max Stress Mpa	Plate Thickness
1	4-Node Shell	10 x 10 mm	no	12.4	12 mm
2	8-Node Shell	10 x 10 mm	no	11.9	12 mm
3	20-Node Solid	10 x 10 mm	no	14.2	12 mm
4	8-Node Shell	25 x 25 mm	no	9.6	12 mm
5	8-Node Shell	10 x 10 mm	no	11.9	5 mm
6	20-Node Solid	10 x 10 mm	yes	12.2	12 mm

elements. This may be due to the stress values being calculated at the element centroid rather than at the surface as they were in the plate elements. When the weld was included in the 20-node model the stress in the element adjacent to the weld toe was 12.2 which was very close to the value for 4 node plate element model.

6.2 Plain Plate with Hole

An analysis of detail 28 of AISC category 28, a plain plate with drilled hole, was carried out to assess the use of 4-node and 8-node plate elements and the 20-node solid element in determining the hot spot stresses when subjected to a tension load. Use was made of bar elements with essentially zero cross section area to acquire the hot spot stress in the plate element models. The geometric stress concentration factor was also determined. The elements around the hole were sized to roughly meet the t and 2t rule. Figure 19 shows the stress distribution for the 4-node plate element model where the nominal stress is 1000 units. The bar elements in the expanded view, showed a maximum stress of 4375 while the maximum in the elements at the hole edge was 3761. The geometric stress concentration is 4375/1000 = 4.375. The ratio of hole diameter to plate width was .5 for which Peterson[13] gives a value of 4.32.

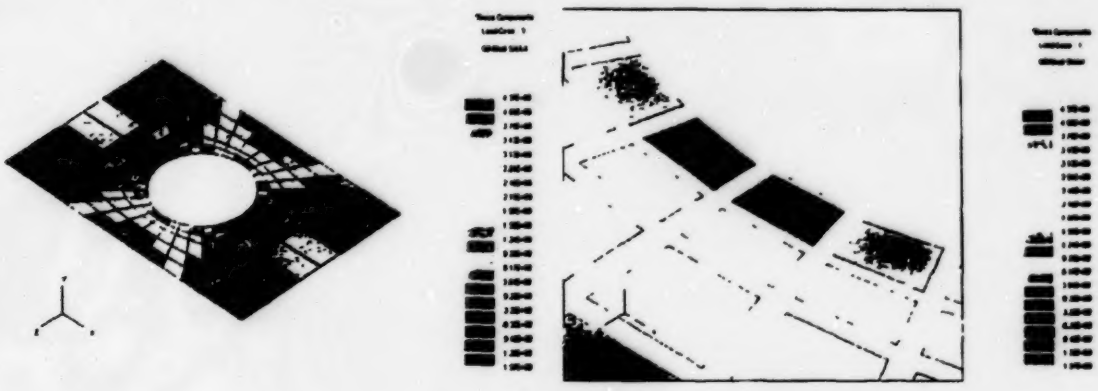


Figure 19: 4-Node Plate and Bar Element Stresses in a Plain Plate with a Hole Under tension

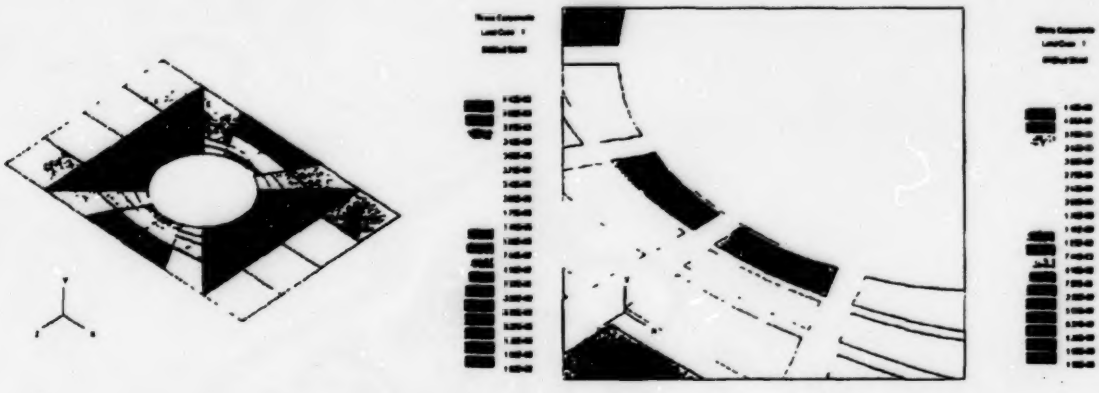


Figure 20: 8-Node Plate and Bar Element Stresses, in a Plain Plate with a Hole under Tension

The results of the plate with a hole, when modeled with 8-node plate elements and bars around the hole edge, are shown in Figure 20. The maximum stress in the plate elements was 3482 and the bars was 4419. The geometric stress concentration in this case is 4.42.

The results of the analysis of the plate with a hole, when modeled with 20-node solid elements, are shown in Figure 21. The maximum stress in the solid elements was 3520 and in the bars it was 4440 giving a geometric concentration factor of 4.44.

A finer 20-node grid model of the plate with a hole was created to assess the affect on the stresses. The finer grid while it significantly increased the maximum 20-node element stress did not appreciably change the maximum stress value in the bars.

The results of the plain plate with hole analysis are summarized in Table 2. It can be seen in the table that the bar elements gave a fairly consistent value for the hot spot stress regardless of the type of element used and the fineness of the grid The maximum bar stress increased by only 4.5 percent compared to a 23 percent increase in the element stress. The results obtained from the 4 models indicate that the use of essentially zero stiffness bar elements with 4-node

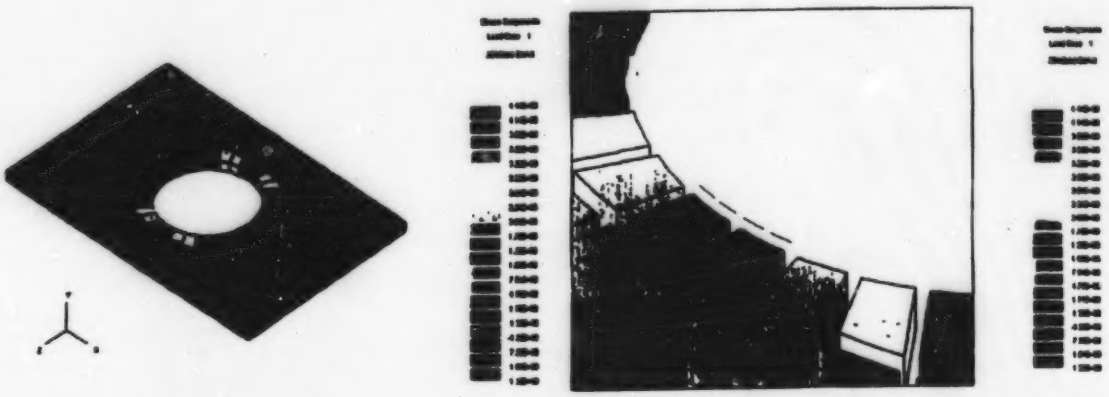


Figure 21: 20-Node Plate and Bar Element Stresses, in a Plain Plate with a Hole under Tension

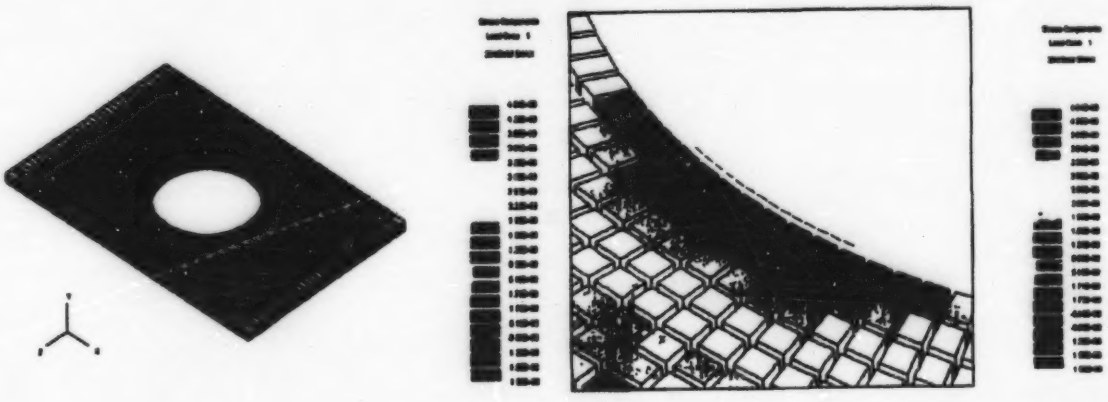


Figure 22: 20-Node Plate and Bar Element Stresses, in a Plain Plate with a Hole under Tension using a Finer Grid

Table 2: Max Element and Hot Spot Stresses in a Plain Plate with a Hole, Models 1 to 4

Model	Element Type	Max Element Stress	Hot Spot Bar	Stress Concentration
1	4-Node Shell	3761	4375	4.375
2	8-Node Shell	3482	4419	4.420
3	20-Node Solid	3520	4440	4.440
4	20-Node Solid	4332	4640	4.640

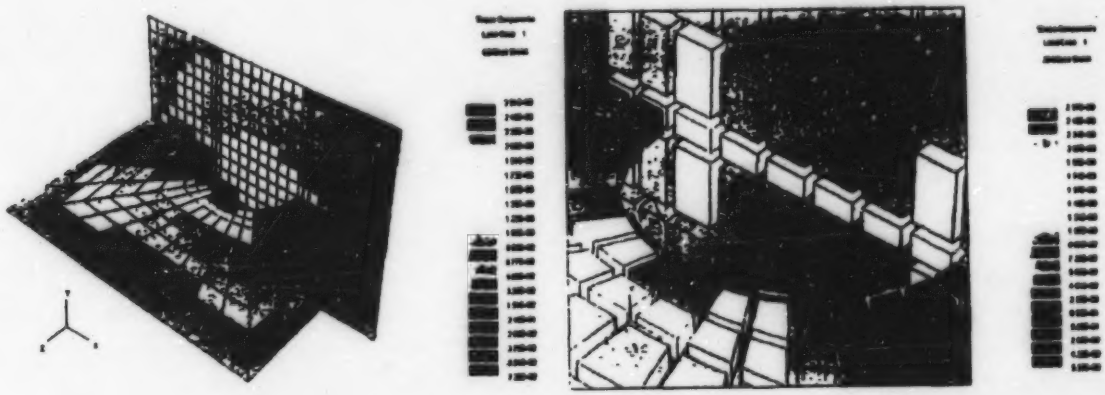


Figure 23: 4-Node Plate and Bar Element Stresses in a T section with a Cut-out under Tension

plate elements is an efficient method for obtaining hot spot stresses in a plain plate with a hole.

6.3 T Section with a Cut-out

A T section, with a semi circular hole, was analyzed to assess the affect of the hole when the section was subjected to tension and and transverse loads. The hot spot stress at the hole was determined by the use of bar elements with near zero stiffness. The first finite element model was discretized with 4 node plate and bar elements at the hole edge. The model was loaded in tension in the X direction to produce a nominal stress of 1000 units. The results of the analysis are shown in Figure 23. The maximum element stress was 2194 and the maximum hot spot stress in the bars at the hole edge was 2610.

A second model was created using 20-node elements along with the bars and subjected to the same load. The results as illustrated in Figure 24 show a maximum element stress of 2062 and a hot spot stress in the bars of 2570.

A third model, shown in Figure 25, was generated to assess the effect of refining the grid of the first model. Under the same loading the maximum element stress increased to 2378. The hot spot stress in the bars stayed essentially the same at 2620. This assessment showed that the use of bar elements with plate elements in the high stress region, with dimensions approximately equal to the plate thickness, gave a good estimate of the hot spot stress and that there was little advantage in using a more refined model. The results of the three models loaded in tension are summarized in Table 3.

The 4-node plate element model was used to assess the effect on the hot spot stress location

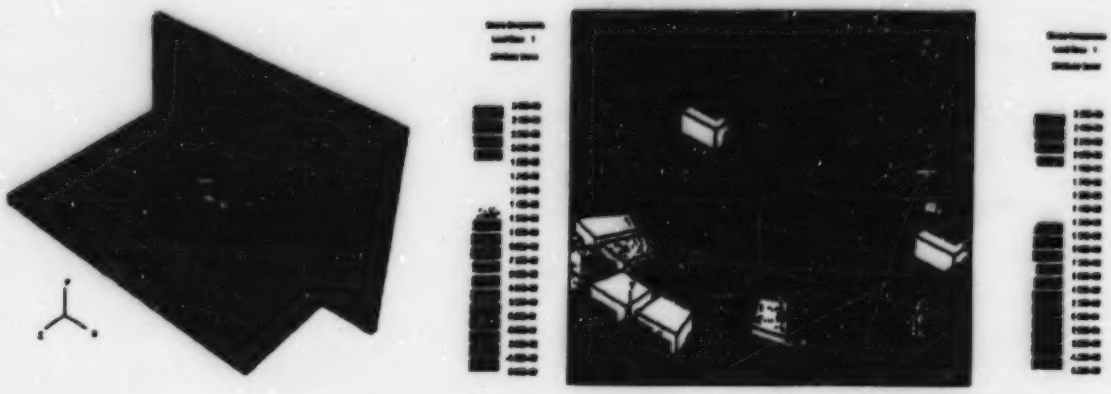


Figure 24: 20-Node Solid and Bar Element Stresses in a T section with a Cutout under Tension

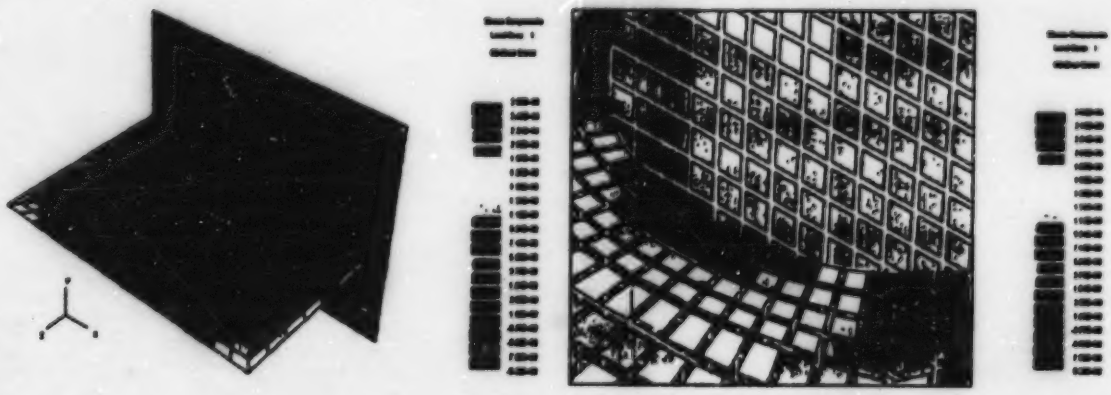


Figure 25: 4-Node Plate and Bar Element Stresses in a Refined T section with a Cutout under Tension

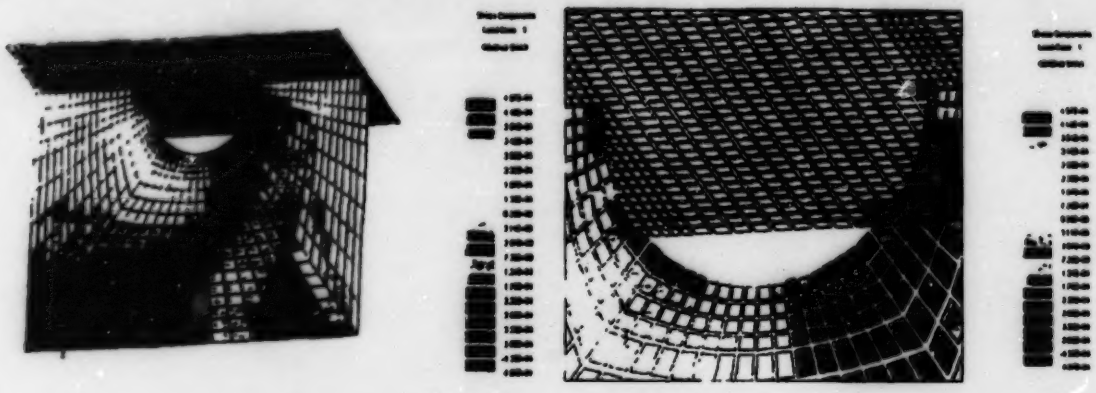


Figure 26: 4-Node Plate and Bar Element Stress Distribution in a T section with a Cutout under a Tension and Transverse Load

Table 3: Max Element and Hot Spot Stresses in a T Section with Cut-out

Model	Element Type	Max Element Stress	Hot Spot Bar	Level of Refinement
1	4-Node Plate	2194	2610	1
2	20-Node Shell	2062	2510	1
3	4-Node Plate	2378	2620	2

caused by a combined tension and transverse load. The tension load was applied in the positive X direction to the the flange and web edges. The transverse load was applied in the positive Z direction to the web edge only. The results of this analysis are shown in Figure 26. The hot spot stress locations as seen in the bar elements are representative for the type of loading.

6.4 A Ship Bracket Detail

A finite element model of a ship bracket detail was generated, as shown in Figure 27, for comparison with a benchmark analysis carried out in Reference[14] to determine hotspot stresses. An attempt was made to duplicate the original model as much as possible. The bracket was connected to a deck and a transverse bulkhead which were modeled as bar elements using $40t^2$ for the cross section area where t is the plate thickness. Bar elements were also used for the flange of the bulkhead stiffener. The deck is represented in Figure 27 by the blue bar elements and the transverse bulkhead by the green bar elements. Bars of essentially zero stiffness were used to pick up the hot spot stresses at the toe of the stiffener plate edge and along the deck beam from the toe. The bracket was subjected to a prescribed displacement of 1 mm in the X direction at the right end and a prescribed displacement of -.5 mm in the X direction at the bottom. The model described in the referenced report was solved using the finite element program ANSYS. The finite element program VAST was used to solve the duplicate model.

The results of the analysis are shown in Figure 28. The stresses in the plate elements and bar element in the referenced report were chosen at 10 mm from the intersection of the bracket gusset and the beam flange. Plots of the hot spot stresses in the web and bracket in

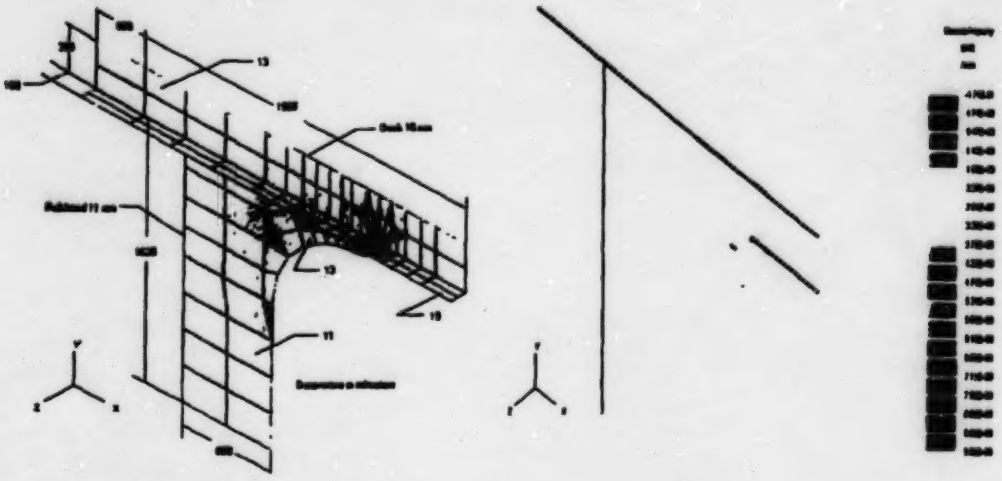


Figure 27: The Ship Bracket Finite Element Model with Bar Sizes and Overall Dimensions

Table 4: A Comparison of Von Mises Element and Bar Stresses in a Ship Bracket Detail

Element Types	ANSYS Original Grid	VAST Original Grid	VAST New Grid
Plate Element Stresses Mpa			
Bracket	209.3	210	202
Longitudinal Web	248.9	268	232
Bar Edge Stresses Mpa			
Bracket	119.8	151	154
Longitudinal Web	235.5	262	237

the duplicate model are shown in Figure 29 and Figure 30 with the stress linearly extrapolated to obtain the hot spot stresses.

An additional model of the ship bracket detail was generated in which the elements in the high stress region were replaced with better proportioned quadrilateral elements. The model and an expanded view of the element stresses are shown in Figure 31.

A comparison of the stresses obtain is made with the published values in Table 4. The agreement between the element results of the ANSYS and VAST analyses was much closer than between the bar element results. This may be partially due to the unsymmetrical bending due to the beam flange not being symmetrical with the web. This resulted in plate upper and lower surfaces having different stress values.

6.5 Cut-out for Stiffeners

Hot spot stresses can be determined in cut-outs such as those made to allow stiffeners to pass through either bulkheads or webs of intersecting stiffeners. The hot spot stresses were obtained by the use of bar elements, with a very small crossection of .001mm^2, placed around

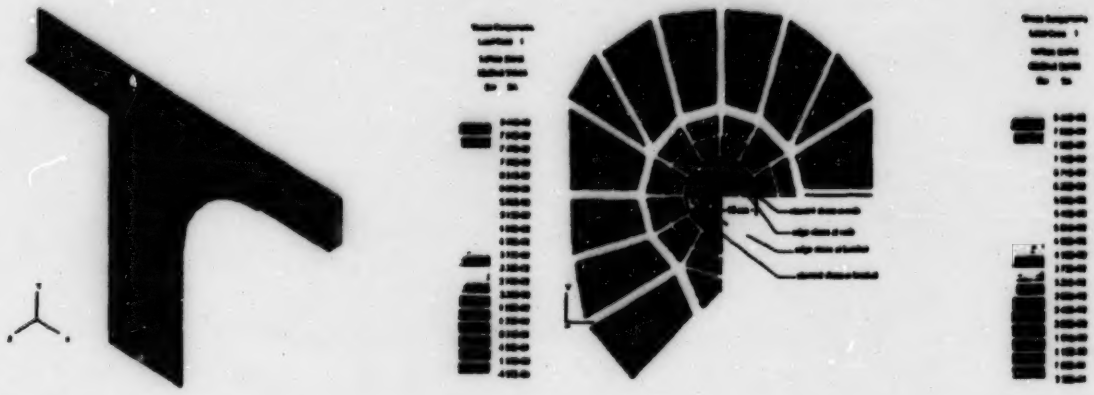


Figure 28: Von Mises Stresses in the Bracket and an Expanded View Showing the Element and Bar Stresses

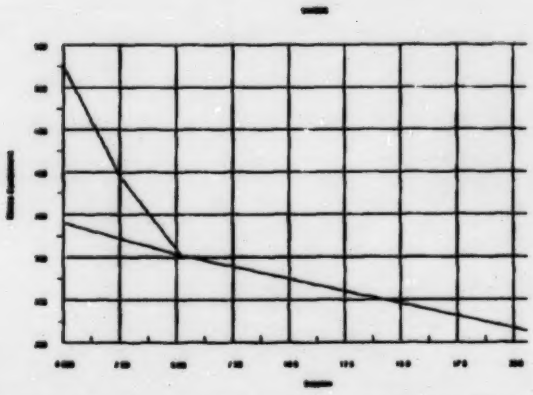


Figure 29: Hot Spot Stress in the Beam Web

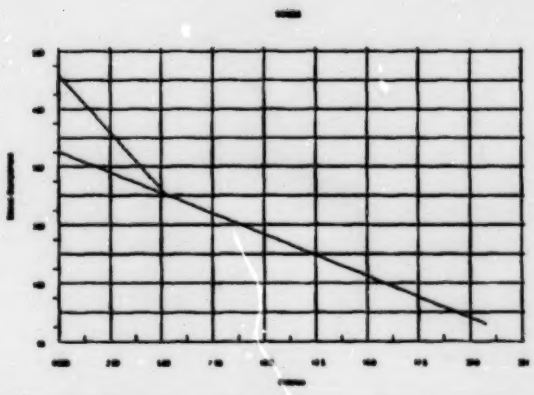


Figure 30: Hot Spot Stress in the Bracket Gusset

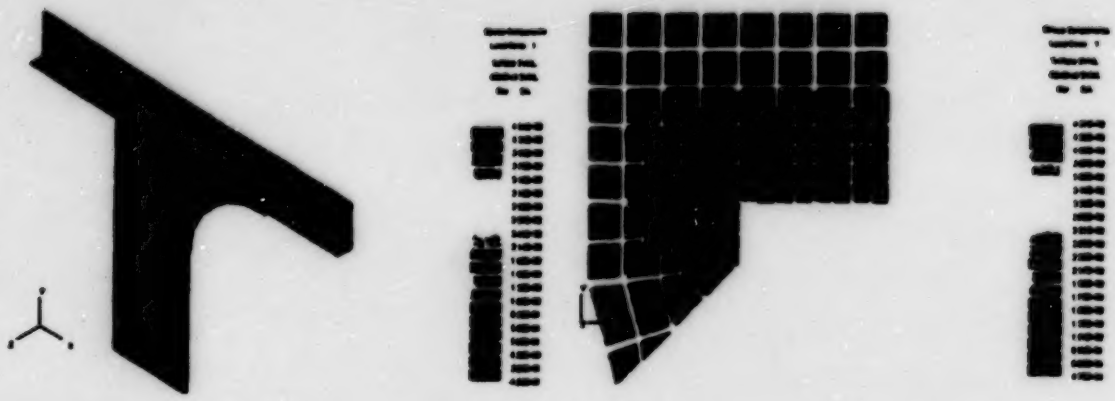


Figure 31: Von Mises Stresses in the Bracket with Better Proportioned Elements and an Expanded View Showing the Element and Bar Stresses

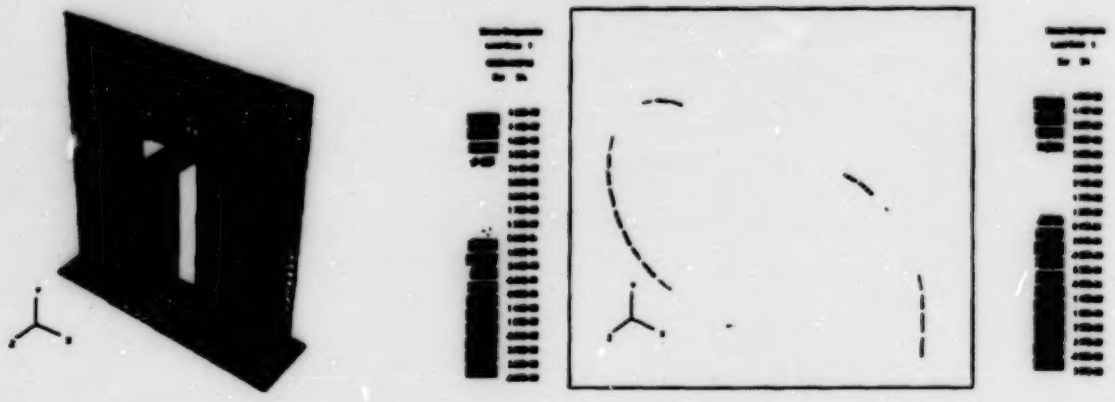


Figure 32: Hot Spot Stresses in a Cut-out Obtained by the use of Bar Elements

the cut-out edge. A finite element model of such a situation is shown in Figure 32. The stress distribution was due to a uniform tension load applied in the X direction at the web and flange edge . The bar elements show a maximum stress of 484 Mpa compared to the value of 492 Mpa in the element whose nodes it shares.

The S-N curve most suitable for this form of cut-out is the B curve of the AISC category of weldments (see Figure 7).

7 Summary and Recommendations

A review was made of methods for obtaining stress values in ship structural details that can be used with the appropriate S-N curve and the Palmgren-Miner's cumulative damage rule to predict fatigue life.

Due to the complexity of most ship structural details, the methods were centered on finite

element analysis to obtain, in the simplest approach, the nominal stress. When the stress distribution was so complex that the nominal stress was not obvious, the hot spot method then became the method of choice. Both the nominal stress and the hot spot methods rely on the availability of S-N data obtained from laboratory testing of typical welded structural details. When this data is lacking for a particular structural detail, then the most exact method for determining the stress in a high stress region is to use an increasingly refined model to determine the maximum stress and perhaps a stress concentration factor for future use. The highly refined model approach was demonstrated using a detailed fine element model of a fillet weld. The model was refined in several stages in the location of the weld toe. To overcome the singularity at the weld toe a 1/32 inch radius was used to successfully achieve convergence and the maximum stress at the weld toe.

The review concentrated on the hot spot approach because of its usefulness and popularity. A number of structural details were modeled starting with a model with the proportions of a fatigue test specimen and equivalent to detail 30 of category E of the AISC catalog of weldments. Six models of the detail were generated using 4-node and 8-node plate and 20-node solid elements. It was shown for the 8-node element, that element size, strictly based on the plate thickness rule, where the size should be equal to twice the plate thickness, was less important than the grid size. It was also shown that the 20-node solid element model produced a much higher stress value than the plate elements.

The results obtained from the analysis of Detail 28 from the AISC category B of weldments, a plain plate with a hole, showed that hot spot stresses and geometric stress concentration factors can be satisfactorily obtained by the use of 4-node plate elements and essentially zero stiffness bar elements.

The analysis of a T-section with a cutout using 4-node plate and 20-node solid elements under tension and combined tension loads showed that bar elements around the cut-out edge gave a satisfactory prediction of the hot spot stresses without the need for a second level of refinement.

The comparison of a VAST analysis of a MG/DSA generated version of a benchmark model of a ship bracket detail with the ANSYS analysis described in Reference 14, showed fair agreement in the element stress results but poor agreement with the zero stiffness bars. Despite considerable effort to achieve identical models, the agreement between results could not be improved. The modification of the high stress region to better proportioned elements resulted in very little improvement in the results indicating that the distortion of the elements in the original grid was not too severe.

The use of zero stiffness bar elements around the opening in the example of a cut-out of a beam web (for a stiffener to pass through) was shown to be an acceptable approach for identifying and obtaining hot spot stresses at the plate edge.

Two element problems were identified from the review. One was that the 8-node plate element did not seem to be as sensitive to the 2t rule as expected. This should be investigated to see if it is related to the 8-node element in VAST. The post processing of the VAST 20-node elements stresses by MG/DSA needs the option to extrapolate stresses from the Gauss points to the surface. At present the stresses are calculated at the centroid of the element which does not give the necessary stresses at the element surface when bending is present. Another difficulty encountered was in identifying the correct S-N curve to use with the hot spot stress obtained from a particular ship structural detail. A further search of the literature should be made to determine the availability of S-N curves, suitable for general ship structure details, that can be directly applied to hot spot results.

References

[1] "MAESTRO-Method for Analysis Evaluation and Structural Optimization, User's Manual Version 6.0", distributed by Ross McNatt Naval Architects, Annapolis, MD., July 1992.

[2] 'MG/DSA' Martec Ltd.

[3] Miner M.A.,'Cumulative Damage in Fatigue', Journal of Applied Mechanics, Vol. 12, 1945.

[4] "Stress Determination for Fatigue Analysis of Welded Components", The International Institute of Welding, Edited by Erkki Niemi.

[5] Cramer E, Gran S, Holtsmark G., Lotsberge I., 'Fatigue Assessment of Ship Structures', Det Norske Veritas, DNVC Report No. 93-0432, Hovik, Norway, Feb. 11 1994

[6] Munse W.H., Wilber T.W., Tellalian M.L., Nicoll K. and Wilson K.,"Fatigue Characterization of Fabricated Ship Details for Design" Report No. SSC 318, Department of Civil Engineering, University of Illinois, Urbana-Champaign, Sponsored by Ship Structure Committee, 1983.

[7] Stambaugh K.A.,Leeson D.H., Lawerence D., Hou C-Y, Banas G.,"Reduction of S-N Curves for Ship Structural Details" Report No. SSc-369, Columbia Research Corp.,Arlington, Virgina, Sept. 1992.

[8] "Fatigue Strength of Welded Ship Structures",Bureau Veritas, NI 393 DSM R00 E, Paris, Sept., 1994.

[9] "Vibration and Strength Analysis Program (VAST): User's Manual", Martec Ltd. Halifax, Nova Scotia, 1997.

[10] "Quasi-StaticResponse" Proceedings of the 13th International Congress Ship and Offshore Structures Congress Volume 1,Edited by T Moan and S.Berge, 1997.

[11] "Fatigue and Fracture" Proceedings of the 13th International Congress Ship and Offshore Structures Congress Volume 1,Edited by T Moan and S.Berge, 1997.

[12] Chalmers D.W., "Design of Ships Structures",UK. Ministry of Defence, HMSO, London 1993.

[13] Peterson R.E.,'Stress Concentration Factors',John Wiley and Son, 1974.

[14] Basu R.I., Kirkhope K.J., Srinivasan J., "Guidelines for Evaluation of Ship Structural Finite Element Analysis",Ship Structure Committee, Report No. SSC-387, 1996.

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1 ORIGINATOR (the name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.)
Defence Research Establishment Atlantic
P.O. Box 1012
Dartmouth, Nova Scotia, Canada B2Y 3Z7

2 SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable).
Unclassified

3 TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title)
A Procedure
Review of Detail Meshes for Application to Fatigue Analysis

4 AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)
D.R. Smith

5 DATE OF PUBLICATION (month and year of publication of document)
June 2000

6a NO. OF PAGES (total containing information. Include Annexes, Appendices, etc)
27

6b NO. OF REFS (total cited in document)
14

7 DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered)
DREA Contractor Report 2000-082

8 SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include address)
Defence Research Establishment Atlantic
P.O. Box 1012
Dartmouth, N.S. B2Y 3Z7

9a PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant)
1gc-11

9b CONTRACT NO. (if appropriate, the applicable number under which the document was written)
W7707-9-7048

10a ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document.)
DREA CR 2000-082

10b OTHER DOCUMENT NOs (Any other numbers which may be assigned this document either by the originator or by the sponsor.)

11 DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)
(X) Unlimited distribution
( ) Defence departments and defence contractors; further distribution only as approved
( ) Defence departments and Canadian defence contractors; further distribution only as approved
( ) Government departments and agencies; further distribution only as approved
( ) Defence departments; further distribution only as approved
( ) Other (please specify):

12 DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected)
Full, unlimited

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

13. ABSTRACT (a brief and factual summary of the document It may also appear elsewhere in the body of the document itself It is highly desirable that the abstract of classified documents be unclassified Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U) It is not necessary to include here abstracts in both official languages unless the text is bilingual)

Methods for determining the stress range in regions of high stress in ship structural details are described with emphasis on the hot spot method for fatigue analysis. Finite element models of typical details were created and analyzed to determine the hot spot stresses at the weld toe. Models of details with cut-outs were also analyzed by using essentially zero stiffness bar elements to determine the hot spot stresses at the plate edges of the cut-outs.

14 KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document They should be selected so that no security classification is required Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included If possible keywords should be selected from a published thesaurus e g Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title)

Fatigue
Finite element
Ship structures
Hot spot method

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM